ECSE 2920: Design Methodology

Deliverable 5

Group 11

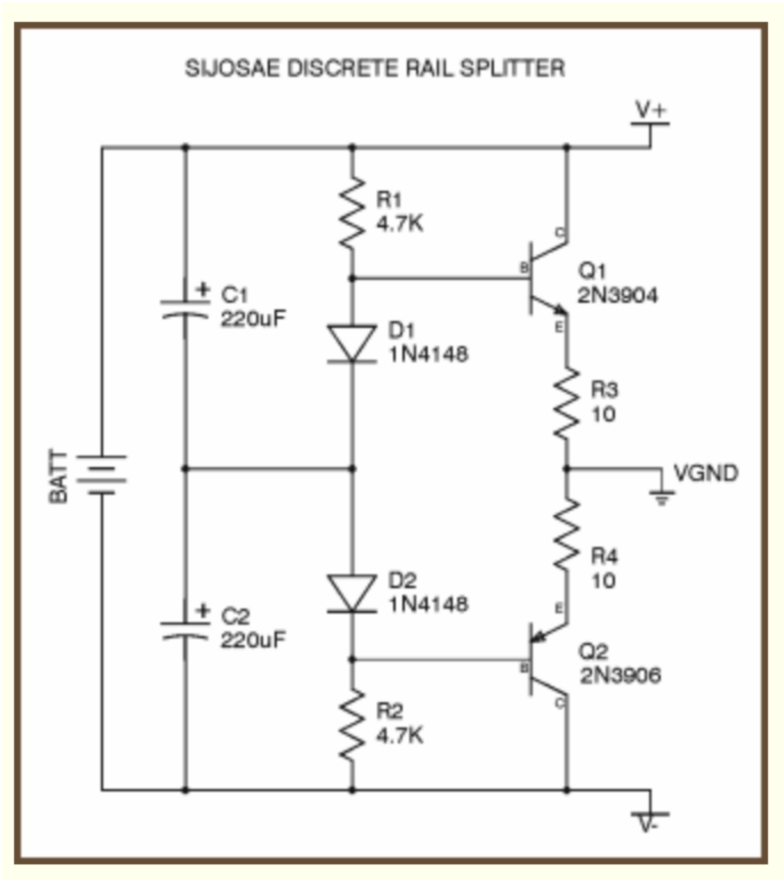
02/05/2025

<u>Part 1: Push/Pull Amplifier</u>

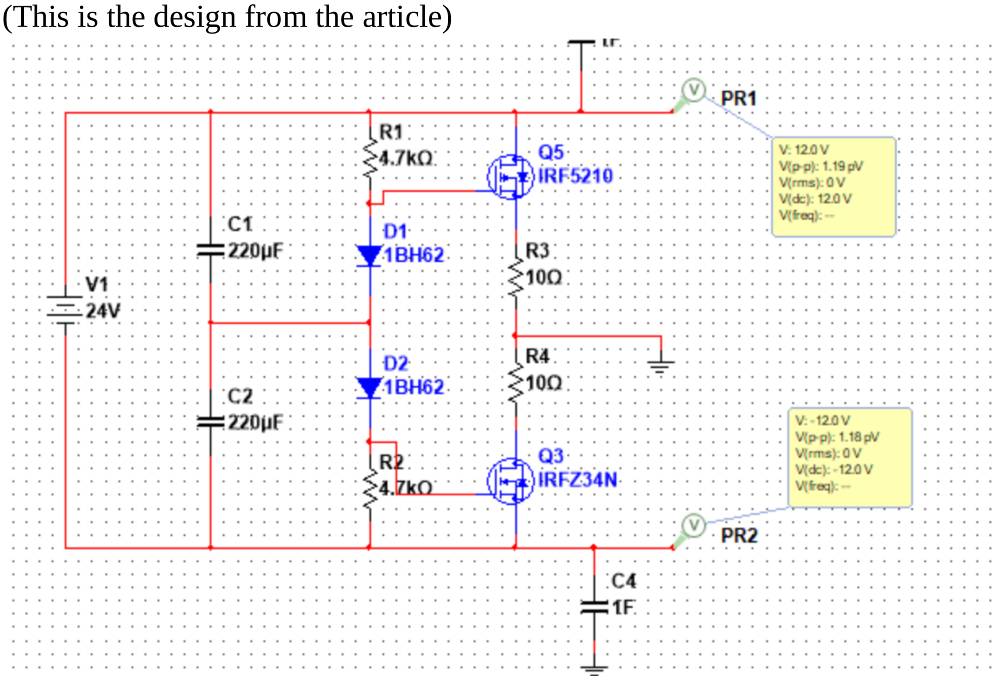
Requirements: Create a simulation of the push/pull amplifier. Include the theoretical design, the simulation (and results) and analysis of the design

Our team took inspiration from an online article titled V*irtual Ground Circuits*. It had a list of different designs with virtual grounds. We decided to opt for the Sijosae discrete rail splitter for a multitude of reasons. For one there were no op amps involved in the design, this is crucial for our case because we are using our push/pull amplifier to power op amps, so logically this would not work. Our design is a buffered virtual ground circuit meaning that the virtual ground is obtained using a buffer. This is crucial because you can make a virtual ground circuit just using a voltage divider, but when you attach a load the resistance inside the voltage divider changes and so does the ratio of voltage division. The buffer fixes this issue. In our circuit the buffer is made using a combination of discrete components (diodes, transistors, and resistors). We altered the design of the circuit from the article slightly because they used BJT transistors, and we are using mosfets, but this is a simple switch. Our team also added bypass capacitors to the +/- rails because the switching of the transistors will cause the voltage at the rails to oscillate. A bypass capacitor will create a stable DC voltage because it will filter the AC signal (the oscillations). This is because in DC the capacitors are just open circuits, so the ground does not short the rail. However, in AC analysis using the phasor domain the impendence of a capacitor is given by 1/jwc, so at a high C value essentially creates a short to ground for AC signals.

Picture(s)

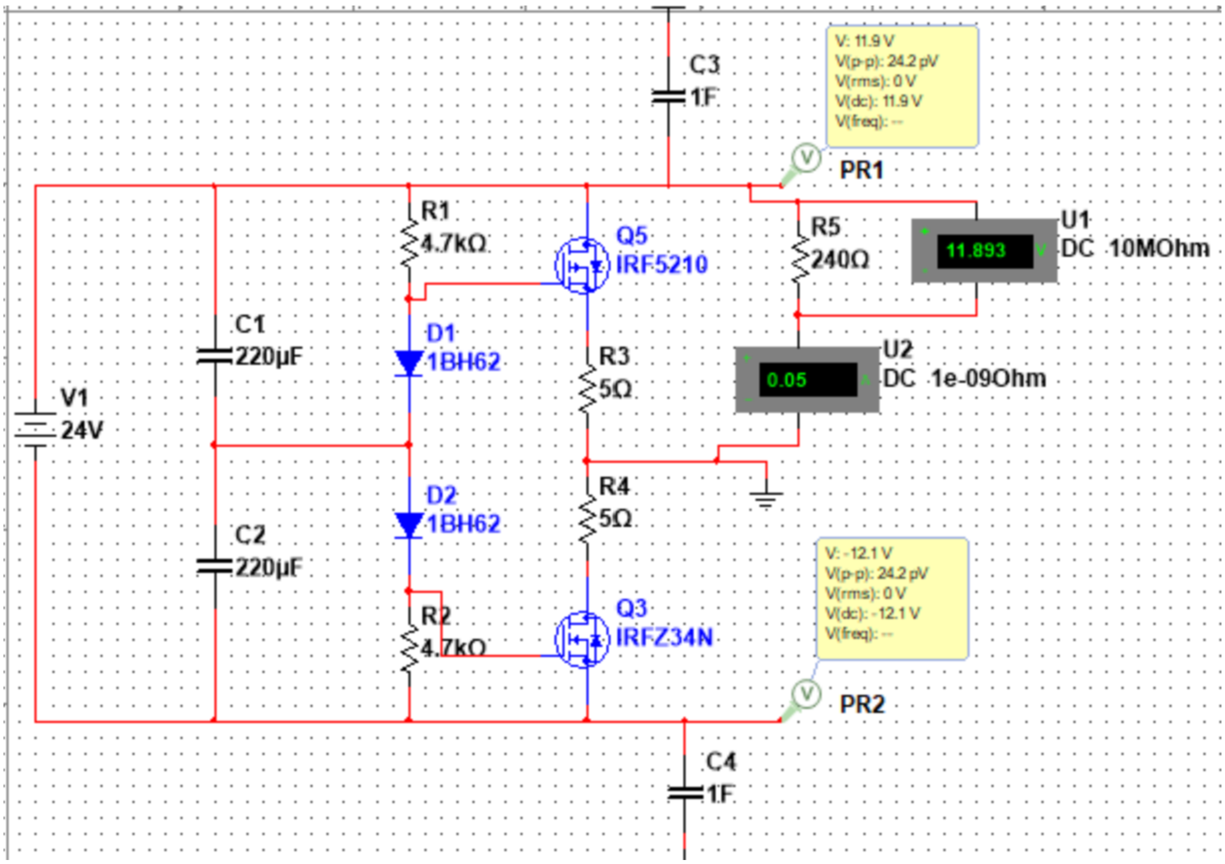


(This is the design from the article)

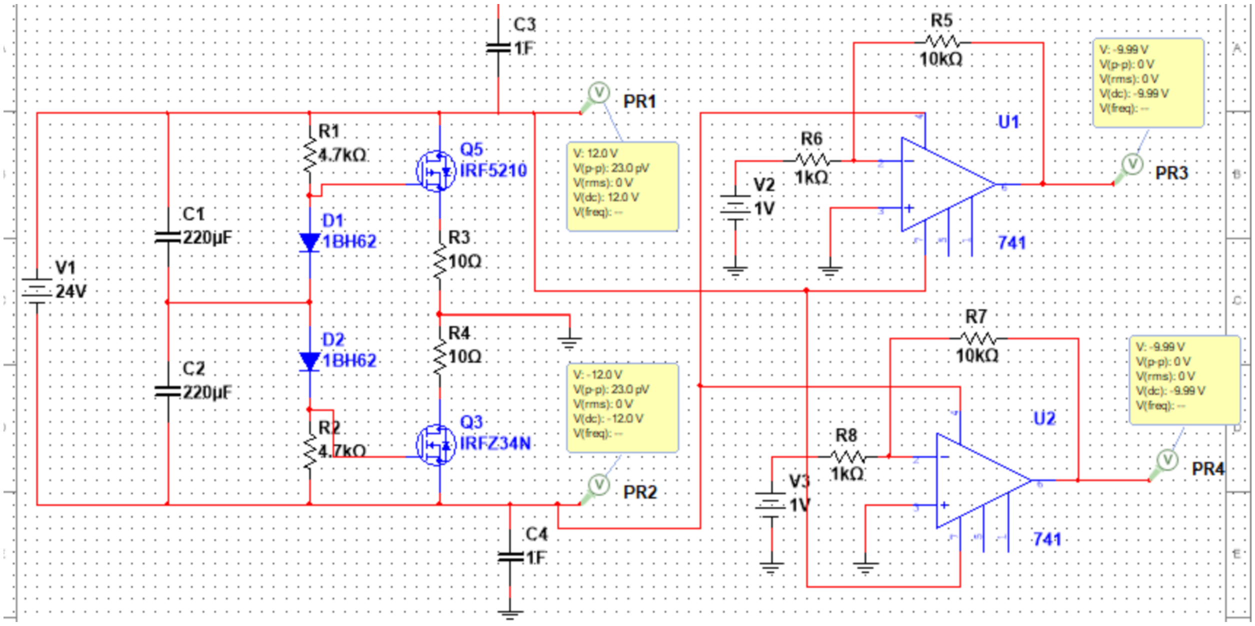


(This is our altered design with the mosphets and bypass capacitors)

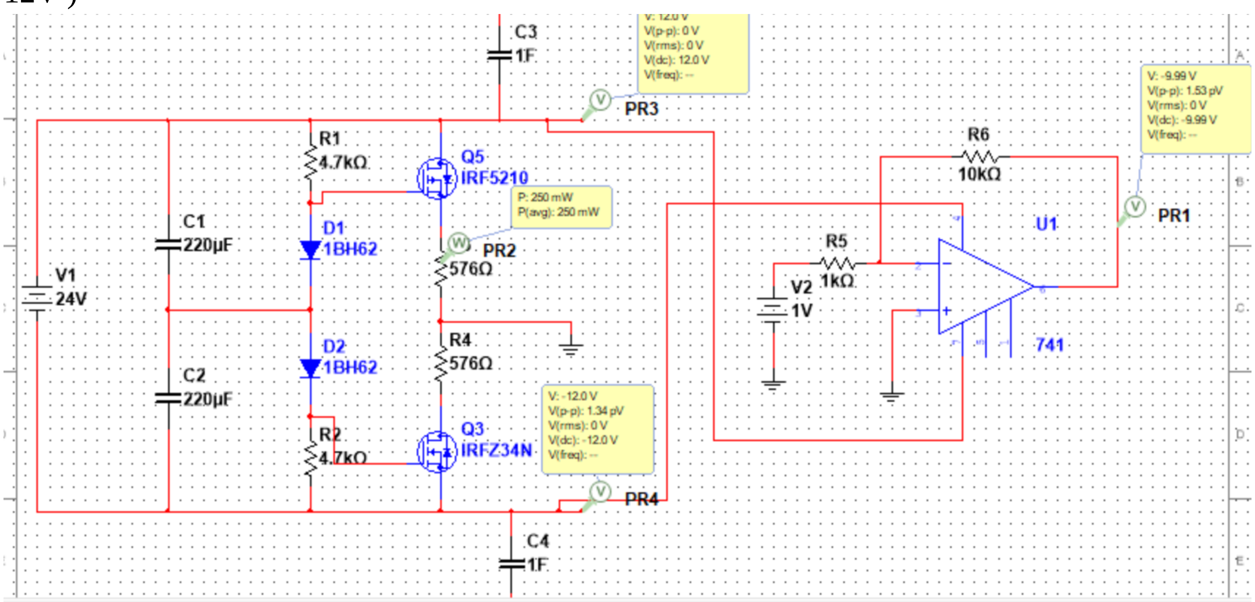
- You can see that our design is stable outputting +/- 12V at the rails without a load



(Here is our circuit with a load with a current of 50mA flowing through it. The voltage at the rails changes by .1V. Which may seem bad put in hindsight 240 Ohms is a very small load and because we are just using the voltage rails to power op amps and ADC, it is okay that the voltage at the rails fluctuates a little bit. This is because the op amp output is limited by the +/- 12, but because we are just using it to power op amps and ADC, we should stay under this anyway)



(Here you can see our push pull amplifier powering 2 inverting op amps as the rails maintain +/- 12V)



(Here we altered the 10 Ohm resistors and changed them to 576 Ohms, in theory this would make the voltage division part of our push pull worse, however because we are using this to power op amps that one, have ideally infinite input impedance, and two are pulling from both rails (still splitting voltage evenly) this change should hardly be noticeable. The reason we want to make these resistors bigger is to limit the current through them. As you can see, they are drawing ¼ Watt which is the max for most resistors in the lab. If we lower the ohms even a little bit, we will probably have to buy high wattage resistors)

Key Design Decision(s)

- What did your team clarify about the design? We made sure to clarify which transistors we were going to use as well as resistor values to divide the voltage evenly to the +/- voltages. We also created our virtual ground using a buffer altering our voltage divider to the right numbers.
- What were the competing choices in the design? At first, we wanted to try and use an op-amp to create the push pull amplifier, but we were not sure if we needed one to get the 12v values we needed. So we had two people work to design one with and without an op amp to see which one would be simpler and more efficient.
- Ultimately what did your team choose and why? We decided to go with the design that did not use an op-amp because it was simpler, and we got it to work testing the physical circuit. We referenced this design with the photo above called the sijosae rail splitter.

TEST… Test… test

- What aspects of the design need to be tested?
    - Software? We created and tested our design in Multisim which got us the +/- 12V values that we needed. We made sure to select the specific parts that we had physically in the Multisim such as our mosfets and diodes. This would allow us to proceed with physical testing of our amplifier.
    - Hardware? We created our circuit in Multisim on a breadboard and tested the values of each voltage to make sure our design was working. We then connected our +/- 12V circuit to an op amp with a gain of –10 to further prove our design worked.
- Who is responsible for testing?
    - Tests ran? We tested our design by plugging our +/- 12V push pull amplifier into a simple inverting op amp with a gain of –10 to see if we could power the op amp properly. We used 1V DC power and according to our Multisim we should be getting around –10V as the output.
    - Conclusions from testing? When testing with the op amp, we ended up getting the –10V output that we were looking for which proved our design was working. We also made sure to check the voltage values of our push pull amplifier to confirm each end was +12 and –12 volts before powering the op amp.

summary/part conclusion (make sure you address all parts of the requirements)

- In conclusion, we learned how to quickly research and create a push pull amplifier. The push and pull out of each transistor to the load allows the circuit to have +/- voltages. We took advantage of this by referencing the sijosae push pull amplifier and editing the circuit to fit our specific design of powering both rails to +/- 12V. We were able to use multisim to complete our design and successfully recreate the circuit physically using a breadboard. Using the breadboard, we were able to power a simple inverting op amp which proved our design worked and is ready for future use in the project.

Part 2: Component List

Requirements: List all components needed to realize the design and the status (have, ordering, etc).

Resistors: 2x 4.7k ohms, 2x 576 ohms, 1x 1k ohm, 1x 10k ohms (Have)
Capacitors: 2x 220 uF, 2x 1F (Have)
Op-amp: 741 op amp (Have)
24 DC supply (Have)
2x 1BH62 LEDs (Have)
2x MOSFETS (Have)

summary/part conclusion (make sure you address all parts of the requirements)

- We listed all the components needed to realize the design and we actually already have all the components needed for the design. We wrote down all components used in Multisim and went to the circuits lab to collect the parts we needed. This allowed us to test the theoretical design physically with good accuracy since we were able to use the same components.

<u>Part 3: PCB</u>

Requirements: Using the Fusion360 file given:

(a) take a look at the routing required to map all the GPIOS. From the material lectures describe some of the design choices made in routing.

- **Any bend in the board is made with 45-degree angles only. No curves or 90-degree corners.**
- **Whenever a wire needs to cross over others, they enter a different layer**
- **Connection routes are normally extremely direct between components**

(b) Why does the board have multiple layers?

- **The board has multiple layers so that wires can cross efficiently whilst keeping each layer clean and less likely to malfunction.**

(c) What are they and what function do they provide?

- **Top – Contains main routing between GPIO pins on pihat, their diodes/LEDs, and their receiving ends on the bottom layer.**
- **Route 2 – Connects the 3.3V GPIO pins from the female header and sends to external screw terminals. This explains why the male 3.3V pins on the pihat don't supply any voltage.**
- **Route 15 – Connects the 5V GPIO pins from the female header and sends to external screw terminals. This explains why the male 5V pins on the pihat don't supply any voltage.**
- **Bottom - Serves to take any wiring from the top layer and either bridge crossing connections together or bridge a connection from the top layer to its corresponding GPIO pin on the 40-pin female header.**
- **Pads – Displays the location and size of PCB pads on the pihat.**

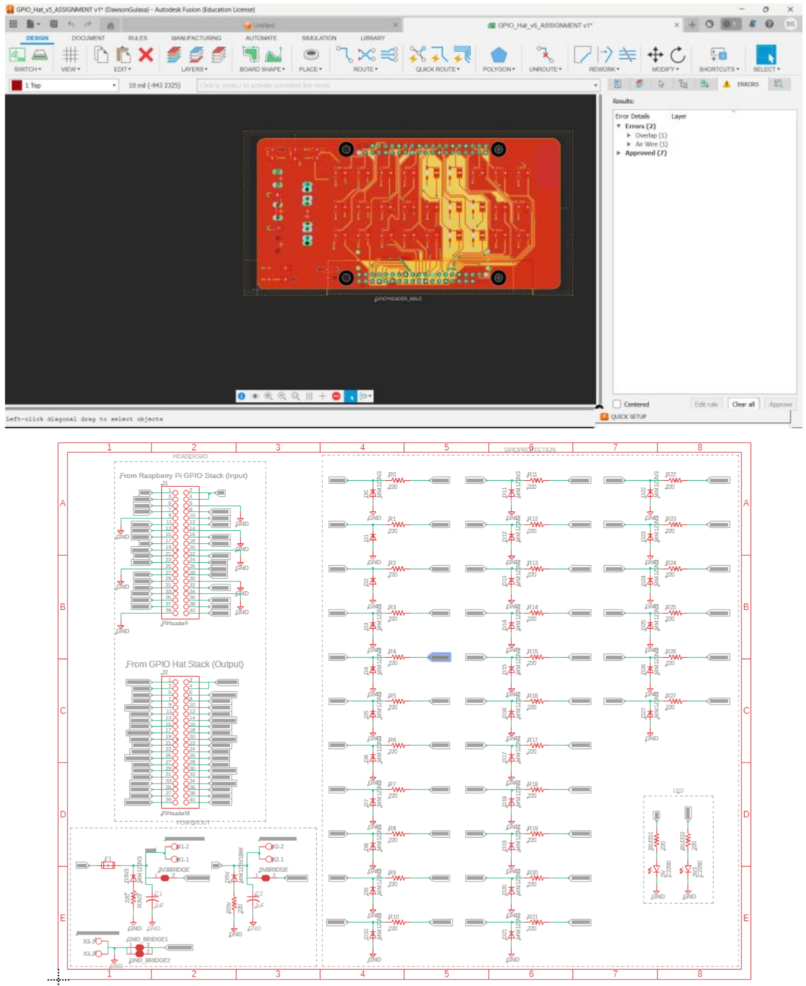
(d) Why are the Zener diodes in the opposite direction of the regular diodes.

- **Because they're meant to be a safeguard. Under proper circumstances, not much current should be passing through the Zener diode and should instead be passing through the resistor and into the GPIO pin on the pi. This is because the normal operating voltage of the GPIO pins is below the Zener voltage on the diode, thus preventing the reverse flow of current through it. When voltage is supplied to a GPIO pin on the pihat that goes above the diode's Zener voltage, the diode allows for the current to pass through it, thus detouring the current straight to ground and stopping it from reaching the pi and possible damaging it.**

(e) Any other comments from analyzing files.

- **Seeing a professional example of a PCB helps me gain a better understanding of what to expect. Unless the design is extremely simple, you shouldn't have one layer containing everything, and it is good practice to not only separate wirings into layers but also make sure that each layer has a purpose in mind.**

Picture(s)



(Showing that the given file is opened in our PCB software program of choice (fusion))

summary/part conclusion (make sure you address all parts of the requirements)

- This deliverable involved diagnosing and analyzing the given PCB (Pi Hat Protector) design. The team clarified routing strategies, layer functionality, and protection measures, particularly focusing on the Zener diodes and GPIO connections. We chose **Fusion360** as the software for PCB design and modification, considering its compatibility and accessibility. Testing of the board confirmed that the design adheres to best practices, including the use of multiple layers, 45-degree bends for traces, and efficient routing for power and signal paths. Moving forward, the team will finalize the Checkpoint B design and plan for PCB fabrication, potentially sourcing from JLCPCB.

Conclusion and Participation (REQUIRED)

1. **Include a selfie/photo from your group meeting/zoom call.**

2. **When did you meet? 1/31 and 1/10**
3. **Who was present? Everyone was present for at least one meeting**
4. **Who was not present? N/A**
5. **What were the main ideas discussed or major decisions (1-2 sentences/bullet points)**

- Push/Pull Amplifier Design: The team evaluated different approaches, ultimately deciding on a non-op-amp rail-splitting push/pull amplifier circuit. The final design utilized MOSFETs, bypass capacitors, and resistors to stabilize the +/- 12V rails for powering op-amps.
- PCB Design and Analysis: We reviewed the provided Pi Hat Protector (PHP) PCB in Fusion360, discussing routing strategies, multi-layer design, and protection features like Zener diodes. The decision to retain the multi-layer structure for routing and separation of power and signal paths was confirmed.